Workshop 8

M2D

8.1 Introduction

This workshop will teach you how to use SMS with the finite difference model M2D. In this workshop an M2D grid will be created. The boundary conditions will be extracted from an ADCIRC simulation. Finally, the M2D solution will be analyzed. For this session, you will need the following files:

- Shinnecock.grd
- fort.63
- Shinnecock.sup (this also uses Shinnecock.xy and Shinnecock.dat)
- coastlinearcs.map
- shinn_AllExtracted.m2c

8.1.1 Extracting Boundary Conditions from ADCIRC Solution

Load the ADCIRC Data and Bathymetry

The ADCIRC solution data will be used to determine the M2D boundary conditions. To load the ADCIRC data:

1. Open the geometry file Shinnecock.grd.
2. Open the solution file fort.63. This is the water surface elevation solution.

The ADCIRC data are in UTM coordinates in the Geographic NAD 83 system. To tell SMS that these are the coordinates for the ADCIRC data:

1. Select *Edit* | *Current Coordinates*.
2. Change the horizontal system to *Geographic NAD 83 (US)*.
3. Change the vertical system to *Local* with units in *meters*.
4. Click *OK*.

The M2D simulation will be performed using State Plane coordinates instead of Geographic coordinates. The ADCIRC data must be converted. To make this conversion:

1. Select *Edit* | *Coordinate Conversions*.
2. In the convert to box, change the Horizontal System to *State Plane NAD 83 (US)* and the zone to *New York Long Island – 3104*. Set the units to *meters*.
3. Change the vertical system to *Local* with units in *meters*.
4. Click the *Convert* button.

The data will now be in State Plane coordinates.

8.2 Creating the Grid

M2D grids are created using the map module along with the scatter module. Scatter points in the scatter module contain bathymetric data for the model. The map module is used to define the grid location, cell sizes, and wet and dry areas of the grid.

For this simulation, a set of scatter points have previously been saved to a file. To load the scatter set:

- Open the file *Shinnecock.sup*.

The scatter points are displayed in a small section of the ADCIRC model. The ADCIRC simulation is on a much larger scale than the M2D simulation. To make the display less cluttered:

1. Open the *Display Options* dialog.
2. Change to the 2D Mesh page and turn off everything.
3. Change to the Scatter page. Turn on the *Boundary* and *Contours* options and turn off the *Points* option.
4. Click the *OK* button.
5. Frame the resulting image inside SMS.

8.2.1 Grid Frame

The domain of the grid is defined within the Map module using an object called a grid frame. All map module objects belong to coverages. To enable the creation of an M2D grid frame, an M2D coverage is required.

1. Switch to the Map module.
2. Choose *Feature Objects* | *Coverages*.
3. Click on the new coverage button.
4. Change the new coverge to be of type M2D.
5. Make sure the *Active* button is checked and click *OK*.

You have just created an M2D coverage and made it active. All map module objects that get created will be part of this new coverage. To create the grid frame:

1. Choose *Feature Objects* | *Grid Frame*.
2. Click the *New Grid* button to create a new grid frame.
3. For the *Grid Frame Origin* use x and y coordinates of 420324 and 68802.
4. Change the *Grid Dimensions* to x and y values of 18900 and 8600.
5. Set the *Angle of rotation* to 22.5 degrees.
6. The specified number of cells does not matter because they will be computed later. Click the *OK* button.

The grid frame should overlap part of the scatter set boundary, as shown in Figure 1.

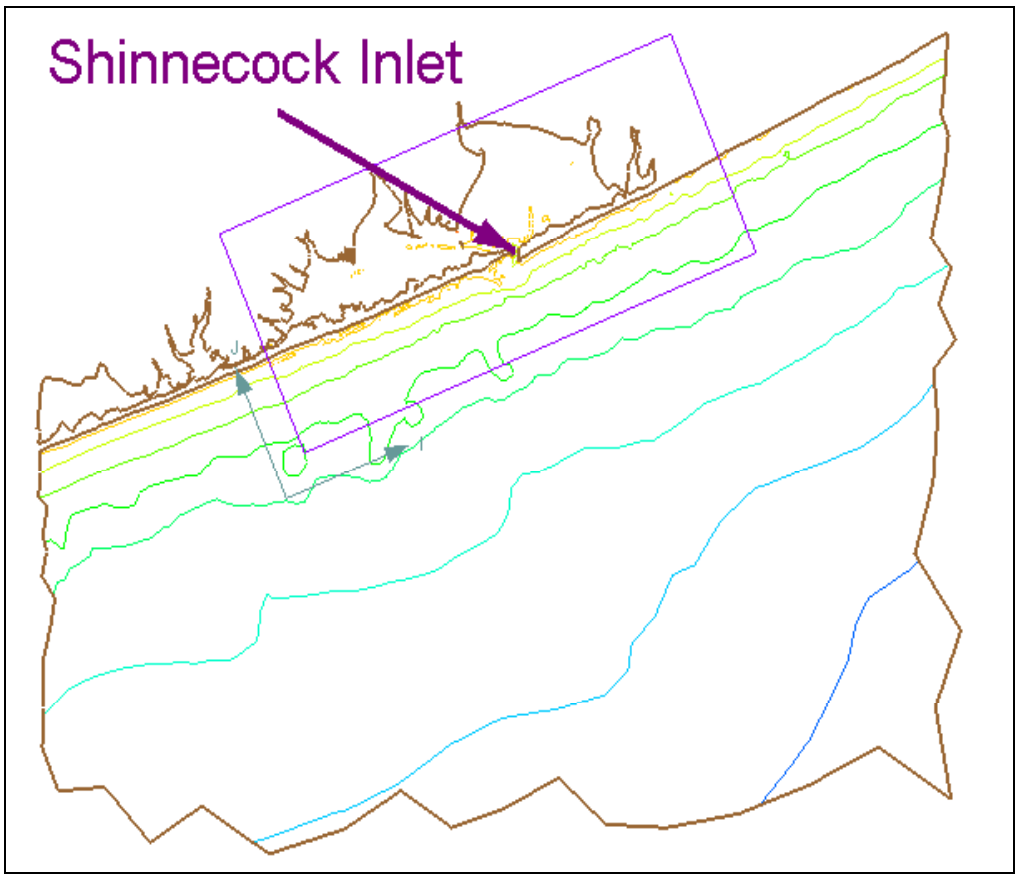


Figure 1 Shinnecock bathymetry and gridframe

8.2.2 Using Refine Points

M2D allows rectangular cells and each row or column of cells can be a different size. The grid cells should be smaller near Shinnecock inlet and can be larger everywhere else. A refine point defines a target resolution at a specific location. SMS will honor the target cell size and provide a smooth transition away from the point. To define a refine point:

- Create a feature point  in the middle of the inlet.

The cell sizes are defined using the point's attributes. To set up the refine point:

1. Select the feature point  and choose *Feature Objects* | *Attributes*.
2. Turn on *Refine grid in the X direction*. Set the *Base cell size* to 35, the *Bias* to 1.05, and the *Max cell size* to 95.
3. Turn on *Refine grid in Y direction* and use the same values as the x direction.

4. Click the *OK* button.

8.2.3 Using Coastline Arcs

Tor perform numerical calculations, the M2D model needs to know the difference between wet (ocean) and dry (land) grid cells. To mark the boundary between wet and dry cells, a coastline arc is used.

For this model, the scatter set boundary can be used to generate the coastline arc. To have SMS create an arc from the scatter set boundary:

1. Switch to the *Scatter* Module.
2. Select *Data* | *Boundary -> Feature*.
3. Return to the *Map* Module.

We only need the arc that defines the coastline and the inlets. To cleanup the arc:

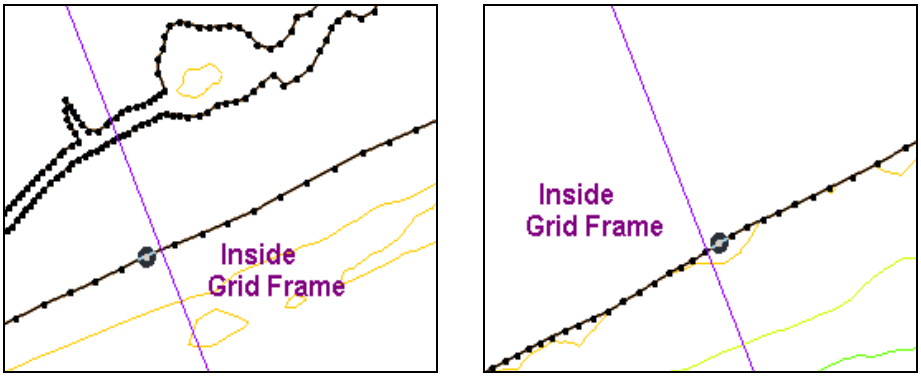
1. Zoom in around the left side of the grid frame.

Select the vertex on the main coastline outside of the grid extents as shown in a. Left side of grid frame. b. Right side of grid frame

2. Figure 2a.
3. Choose *Feature Objects* | *Vertices < - > Nodes*.

Repeat this process for the right side of the grid, as shown in a. Left side of grid frame. b. Right side of grid frame

4. Figure 2b.



a. Left side of grid frame. *b. Right side of grid frame*

Figure 2 Vertices to change to nodes.

Changing vertices to nodes is the way an arc is split into multiple arcs. At this point, the arcs that define the ocean boundary can be removed. To delete them:

1. Select the arcs  that go around the ocean (bottom) boundary of the scatterset. There will be either one or two arcs, depending on where the initial feature point was automatically created (hold the SHIFT key to select multiple arcs).

2. Delete  the arcs.

One of the dilemmas with using coastline arcs is that sometimes they may miss features. If the cells under a coastline were always land cells sometimes skinny bay entrances may be sealed off. If the cells under a coastline were always ocean cells sometimes a jetty would not appear in the numerical representation.

SMS provides three options for coastline arcs to ensure features are well represented by the numerical model. The first option is to make the cells ocean or land based upon the relative areas of the land vs. ocean parts of the cell. If the cell is mostly on land, it will be a land cell otherwise it will be an ocean cell. The second option is to force cells beneath the coastline arc to be land cells. This is useful to ensure that features like jetties are represented in the grid. The third option is to force cells beneath a coastline arc to be ocean cells. This ensures that narrow channels will at least be wide enough for flow to pass through them avoiding isolated puddles.

For the most part we want the grid cells to be determined by percentage of land/water in the cell. Therefore we will make the coastline arc a percentage based arc. To do this:

1. Select the coastline arc.

2. Choose *Feature Objects* | *Attributes*.

3. Change the arc type to *Percent Preference Coastline Arc*.

4. Click *OK*.

When a coastline arc is selected, two arrows are drawn from each end of the arc. One arrow is brown indicating the direction of land, the other is blue indicating the direction of ocean. You must be sure these are facing the right direction. If they are not correct, the arc direction should be reversed. To check this:

1. Select the arc once again.

2. If the blue arrows are pointed upward, reverse the arc by selecting *Feature Objects* | *Reverse Arc Direction*.

Now let's make sure the jetties are not replaced by water cells when creating the grid. We will do this by making the inlet side of the jetty boundary land preference. To force the jetties to be land preference:

1. Zoom the view close to the inlet.
2. On the right jetty, select the vertex nearest to where the jetty begins.
3. Holding down the shift key, select the vertex at the tip of the jetty.
4. With these two vertices selected choose *Feature Objects* | *Vertices <> Nodes*.
5. Select the newly split arc that runs along the jetty.
6. Choose *Feature Objects* | *Attributes*.
7. Set the type to be *Land Preference Coastline Arc*.
8. Click *OK*.
9. Repeat steps 1-8 for the inlet side of the left jetty.

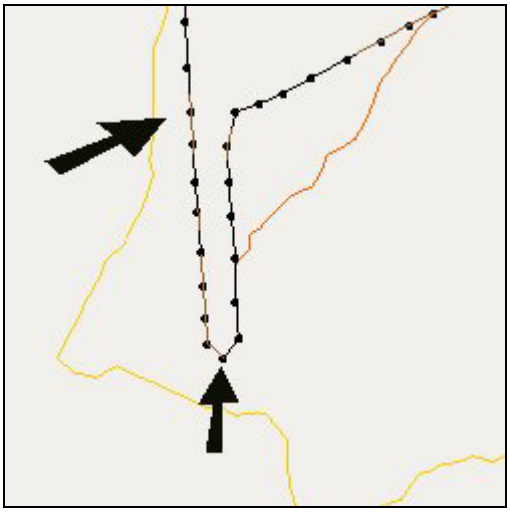


Figure 3 Vertices to select on the right jetty

Now we have protected the jetties from being lost from our grid. The next thing we need to do is make sure that we don't lose narrow channels or bays connected with narrow channels. There is a bay on the right side of the grid that has a narrow opening. To make sure this channel stays wet:

1. Zoom in around the channel as shown in Figure 4.
2. Along the top of the channel select a vertex outside the bay and a vertex inside the bay (hold down the shift key to select both at the same time).

3. Choose *Feature Objects* | *Vertices <> Nodes*.
4. Select the arc that was formed between the selected vertices.
5. Choose *Feature Objects* | *Attributes*.
6. Set the type to be an *Ocean Preference Coastline Arc*.
7. Click *OK*.

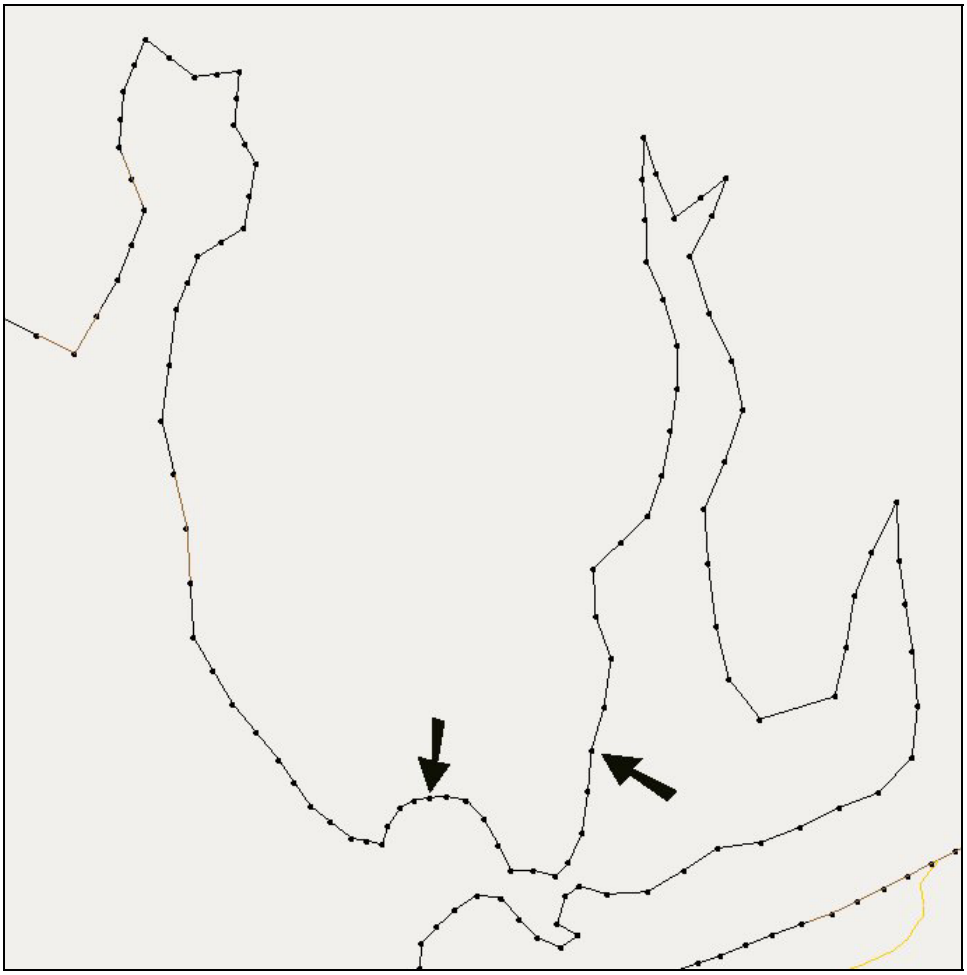


Figure 4 Starting and ending location of ocean preference arc

This gives you an idea how you can use land, ocean, and percent preference coastline arcs so that your grid will best represent reality. Use of this tool requires judgment as to which type of arc to use at different locations. You will now open a map file where all of the arcs have already been split and have been assigned ocean, land and percent preferences.

- Open the file "coastlinearcs.map." A new coverage will be created from the data of the file and will become. This has the refine point and arcs set up.

8.2.4 Map to 2D Grid

Now that the feature and bathymetric data have been prepared, the grid can be generated. To build the grid:

1. Select *Feature Objects* | *Map -> 2D Grid*.
2. The default options should be set correctly, so click *OK*.

SMS will then build the grid.

1. Switch to the *CGrid Module*.
2. Turn off the display of map and scatter entities to reduce clutter.

8.3 Configuring M2D

Once the grid has been created, model parameters and boundary conditions are needed for setting up the M2D simulation.

8.3.1 Setting up the M2D Model Control parameters

We need to setup the model parameters for M2D. These parameters include how long we want the simulation to run, the time step size, and output control. All of these parameters are setup in the model control dialog.

- Choose *M2D* | *Model Control* to open the *M2D Model Control* dialog.

The first options we want to setup deal with the timeframe for our simulation. We want to run our simulation for 72 hours with a one hour ramp. To set the time control options:

1. Click on the *Time Control* button.
2. Change the *Simulation Duration* to 72.0 hours.
3. The *Ramp Duration* should be left at 1.0 days.
4. In the Time Step Size box click on the compute button. This will estimate from the grid geometry the largest timestep that will likely converge. From experience we know that this is too large of a timestep. Change the timestep size to 0.5 seconds.
5. Click *OK* to exit the Time Control Dialog.

The other category of options we need to set deal with the output options for our solution data. We want to output water surface elevation and velocity information for

every cell every fifteen minutes from hour 24 through hour 72. While still in the model control dialog, do the following:

1. We want to output both the velocity and water surface elevation data. Click on the toggle buttons under both of these options in the Output Files box.

2. In the Optional Input Files box, turn on the *Time Specification File for Vector Plot Output* and the *Time Specification File for Global Elevation Output*.

3. To the right of Time Specification File for Vector Plot Output click on the *Choose Time* button.

4. We will add all the times at once. Set the start time to be 24 hours, the increment to be 0.25 hours and the number of times to be 192 and click *Add*.

5. Click *OK* to exit the Output Time Specification dialog.

6. To the right of Time Specification File for Global Elevation Output click on the *Choose Time* button.

7. Click on the button, *Copy Vector Plot Times*.

8. Click *OK* to exit the Output Time Specification dialog.

9. Turn off the Radiation Stresses option at the bottom left and then close the M2D Model Control dialog by clicking *OK*.

The M2D gird should have two water boundaries on the edge of the grid. One boundary is the ocean boundary around the bottom of the grid; the other boundary is a narrow channel at the top of Shinnecock Inlet. Both boundaries need to have boundary conditions specified.

SMS provides a way to use solutions from other models as boundary conditions in M2D. For this example we will use the boundary conditions from a larger ADCIRC model. To extract the boundary conditions:

1. Select all the boundary cellstrings by choosing the Select Cellstring  tool and dragging a box around the entire grid.

2. Choose *M2D* | *Assign BC*.

3. In the left hand side of the dialog select *WSE-forcing*.

4. On the right side of the dialog select *Extract from data set*.

5. Click on the *Extract BC button*.

6. In the tree window, select the ADCIRC dataset for water surface elevation.

7. Since the ADCIRC solution is in seconds, make sure that seconds is selected under Time Units to the right of regional model in the spreadsheet.

8. Make sure the Local Model start time is 0.0 and set the end time to 72.5. This will extract data for half an hour beyond the end of our simulation.

9. Click *Extract*.

10. Click on the *View Extract* button.

11. Look at some of the curves by clicking on their toggles in the left side of the spreadsheet.

12. Click *OK* to exit the *Extracted Data* and *M2D boundary conditions* dialogs.

8.4 Running M2D

We must save the simulation before running M2D. To save the simulation and launch M2D:

1. Choose *File* | *Save As*.

2. Enter the filename "Shin_m2d"

3. Click *OK*.

Running M2D can take a long time especially with a case this complicated. This particular simulation would take about 10 hours to run. Rather than wait for M2D to finish end the process and we will load a previously computed solution. However, the process for running the simulation is given here for your reference.

1. If you were going to run the model on a simulation, you would choose *M2D* | *Run M2D*.

2. Be sure the executable filename is correct. If it is not, then click on the file browser  button and select the correct model executable.

3. Click *OK*. At this point, the model would run with the simulation that you just saved. You should not actually run the model at this point.

8.5 Viewing the Results

Before loading the completed solution, we want to delete all of the data. To clear the data and load the completed solution:

1. Choose *File* | *Delete All* and press *Yes* to confirm.

2. Open the file “Shinn_AllExtracted.m2c”.

Spend the rest of the workshop time analyzing the results.